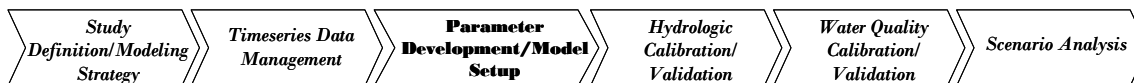




## Exercise 4 – Introduction to WinHSPF and GenScn

### BASINS/HSPF Application Steps



In this exercise, you will learn how to build and run a WinHSPF project. We will add three shape files (*outlets*, *streams*, and *subbasins*) to our BASINS project. These files are associated with the watershed delineation for the West Branch Patuxent River and are necessary to launch WinHSPF from BASINS. They can be created using the BASINS manual delineation tool or automatic delineation tool. Using these files, we will launch WinHSPF from the BASINS interface.

The WinHSPF interface includes many useful tools and user-input windows, many of which allow user-specified parameter adjustments. Because our goal during this exercise is to help you gain a basic understanding of WinHSPF and GenScn, we will leave most of the inputs as the default values. After we have explained each of the tools and some of the input tables, we will run our WinHSPF model for flow.

One of the most important aspects of modeling watersheds is hydrology calibration. In order to ascertain how well our model is simulating conditions in our watershed, we need to compare the model's output to observed data. In this exercise, we will download historical streamflow data, add it to our project's output file, and then use GenScn to plot the simulated streamflow on the same graph as the observed streamflow. In Exercises 5 and 6, we will learn how to calibrate our model's hydrology so that it more accurately predicts streamflow.

### Questions addressed in this exercise:

- 1) How do I import *Subbasins*, *Streams*, and *Outlets* layers into my BASINS project?
- 2) How do I launch WinHSPF from BASINS?
- 3) What tools are available within WinHSPF?
- 4) How do I run my WinHSPF model?
- 5) How do I add gage data to my WinHSPF project file?
- 6) How do I use GenScn to view my simulation results?


### A. Importing delineation layers

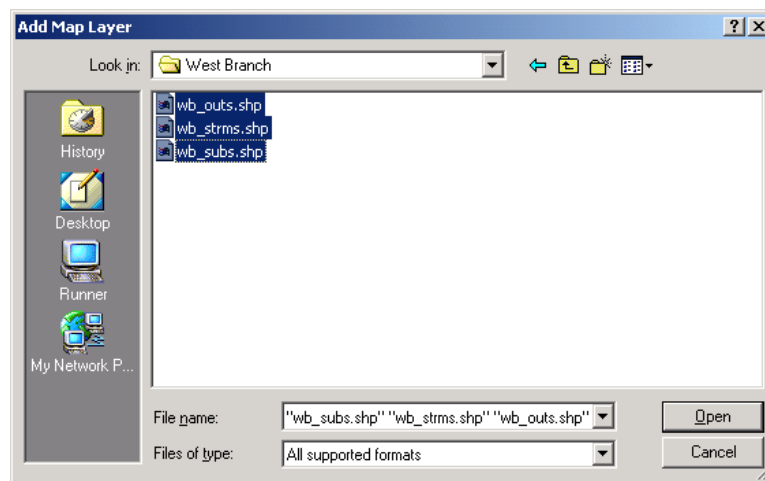
#### QUESTION ANSWERED:

1) *How do I import Subbasins, Streams, and Outlets layers into my BASINS project?*

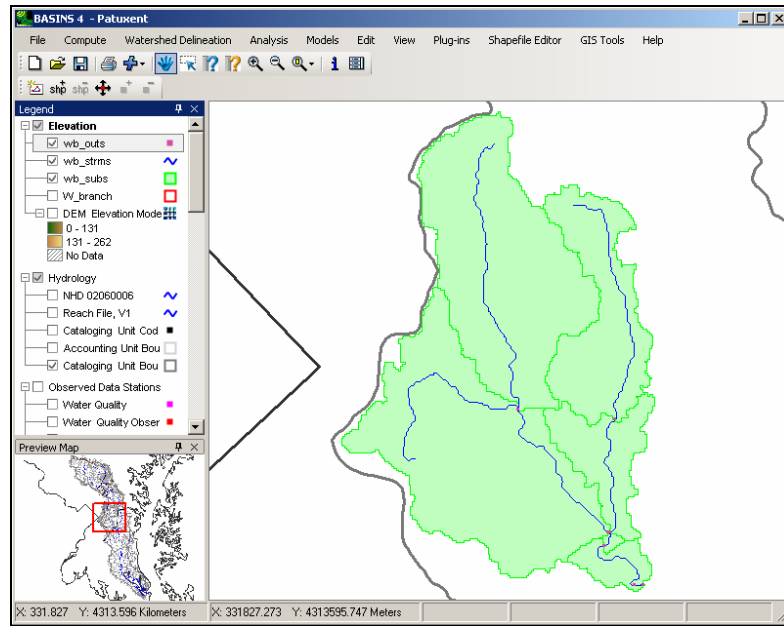
1. If BASINS isn't open, from the **Start** menu under **Programs**, select **BASINS** and then **BASINS 4**.
2. Click "*patuxent*" in the "Welcome to BASINS 4" window.

**Note:** Map layers representing *Streams* and *Subbasins* are required in order to launch WinHSPF from the BASINS interface. An *Outlets* layer is optional. These files can be created using the manual delineation tool or the automatic delineation tool. In this case these files were created by the automatic delineation tool. We will use these predefined map layers so that everybody will have the same model results.

3. Click  (Add Layer tool).
4. Navigate to *c:\basins\predefined delineations\west branch\* and select all three shapefiles in the folder.



5. Click OPEN. Three layers (*Streams wb\_strms.shp*, *Subbasins wb\_subs.shp*, and *Outlets wb\_outs.shp*) will be added to the map.
6. Turn “off” all layers except *wb\_outs.shp*, *wb\_strms.shp*, *wb\_subs.shp*, *State Boundaries*, and *Cataloging Unit Boundaries*.
7. Make sure the *Outlets (wb\_outs.shp)* and *Streams (wb\_strms.shp)* layers are displayed on top of the *Subbasins (wb\_subs.shp)* layer. Your view should look like the following.



**Note:** When layers are loaded into the BASINS map they are assigned an arbitrary color. You can change the color for each of these new layers by double clicking on each layer in the legend and then changing the color in the display properties.

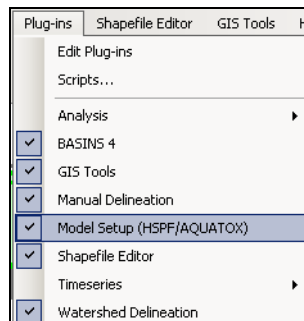
## B. Launching WinHSPF from BASINS


*QUESTION ANSWERED:*

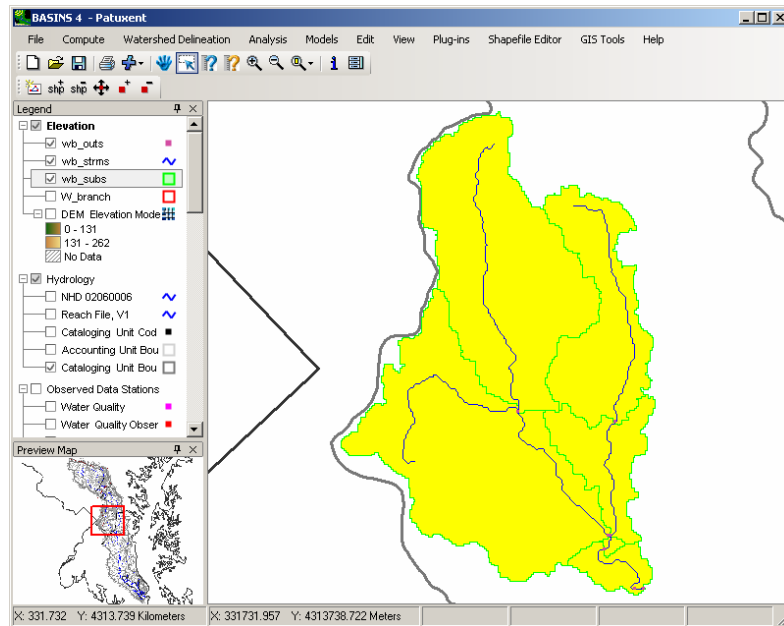
2) *How do I launch WinHSPF from BASINS?*

Now that we have added the new *Subbasins*, *Streams*, and *Outlets* layers, we will launch WinHSPF from the BASINS interface. When WinHSPF is launched from BASINS, an input file (.uci) is created. HSPF uses the information found in BASINS to estimate values for many parameter inputs. It should be noted that these values can be highly inaccurate and should be modified if more accurate information is available.

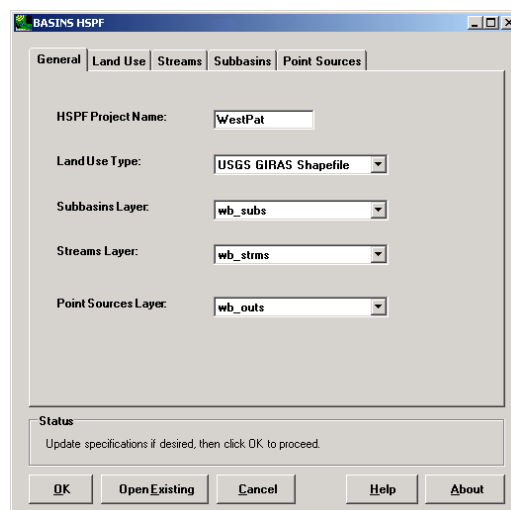
1. From the **Plug-ins** menu, make sure **Model Setup (HSPF/AQUATOX)** is checked (on).



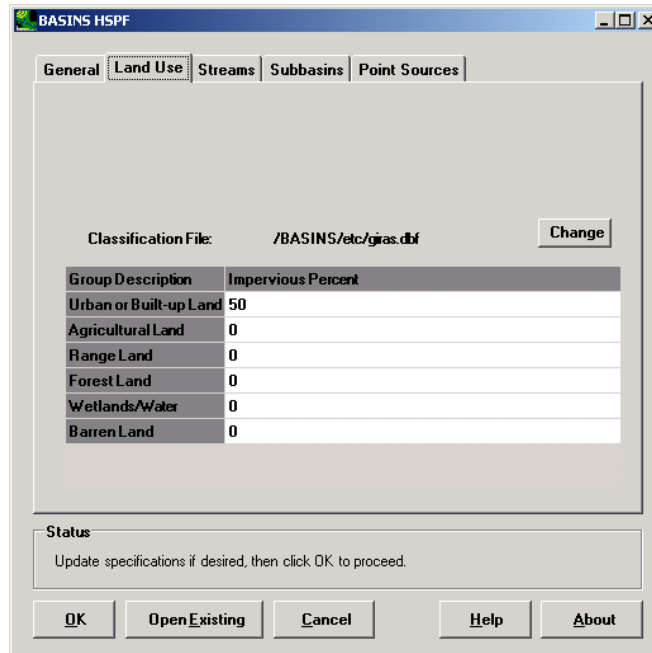
2. Turn “on” and “activate” the *Subbasins* (*wb\_subs.shp*) layer.
3. Click  (Select tool).
4. While holding down the CTRL key, click on all 6 subwatersheds.



5. From the **Models** menu, select **HSPF**.
6. The following window will appear. Enter “WestPat” in the “HSPF project name:” box. Leave “USGS GIRAS Shapefile” in the “Land Use Type” box. Set *wb\_subs* to be the “Subbasins Layer”, *wb\_strms* to be the “Streams Layer”, and *wb\_outs* to be the “Point Sources Layer.”



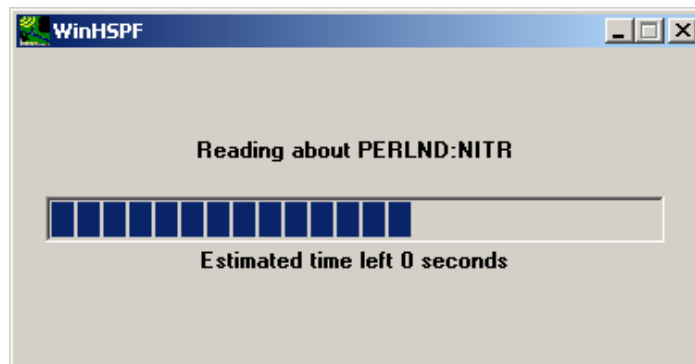
7. Click the **Land Use** tab. The “Land Use” tab is used to specify the land use classification scheme. This scheme includes information regarding the combined land use classifications that will be used in HSPF as well as the impervious percentages for each of the land use classifications.



**Note:** The “impervious percent” values indicate how much of each land use category will be impervious to infiltration, such as paved area and rooftops. For example, a 30% impervious percentage indicates that the given land use category is 30% impervious and 70% pervious. The model uses this information to estimate runoff from each land use type.

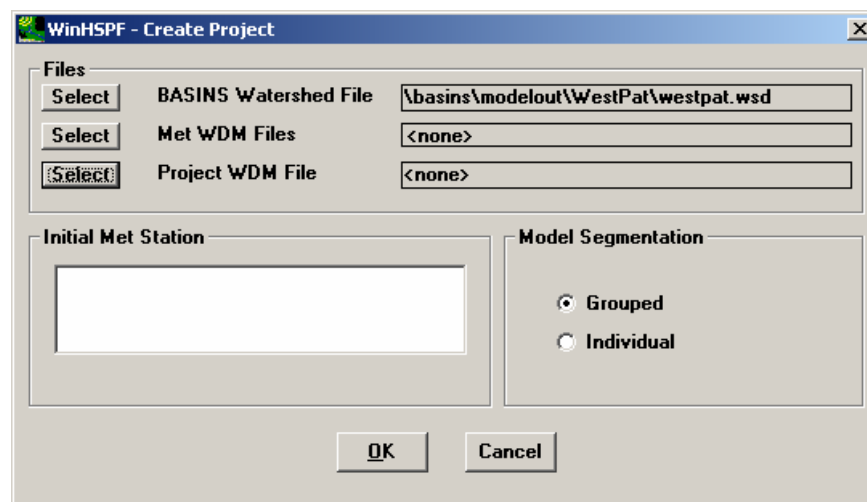
8. Click OK.

**Note:** The following window will appear, indicating that HSPF is initializing.



The “Create Project” window will open. It contains three frames:

- **Files:** This frame includes “Select” buttons for three types of files: the BASINS Watershed File, the Met WDM Files, and the Project WDM file. The BASINS Watershed File field will already be filled in with the name of the \*.wsd file for the current BASINS project that was created when you launched WinHSPF from BASINS. The Met WDM Files ‘Select’ button is used to select one to three meteorological data files. These files include data for precipitation, air temperature, potential evapotranspiration, and other meteorological constituents. Met WDM files are often located in the *c:\basins\data\met\_data\* folder, but any WDM file can be referenced as a met WDM file (as long as the file contains the necessary meteorological data). The “Project WDM” file is created by the user and will contain the point source inputs to the model as well as any output time series from the HSPF simulation.
- **Initial Met Station:** Many WDM Files include data for several stations. In the “Initial Met Station” frame, you should specify the meteorological station that best represents your watershed.
- **Model Segmentation:** Model segments are sub-areas of a watershed with uniform parameters and meteorological inputs that are connected by a reach network. This frame provides two segmentation options that will be explained later.



9. Click the SELECT button beside “Met WDM Files.”
10. Navigate to *c:\basins\data\met\_data\* and select “*md.wdm.*”
11. Click OPEN.
12. Click the SELECT button next to “Project WDM File.”
13. Navigate to *c:\basins\modelout* and type “*w\_branch.*”

14. Click OPEN.

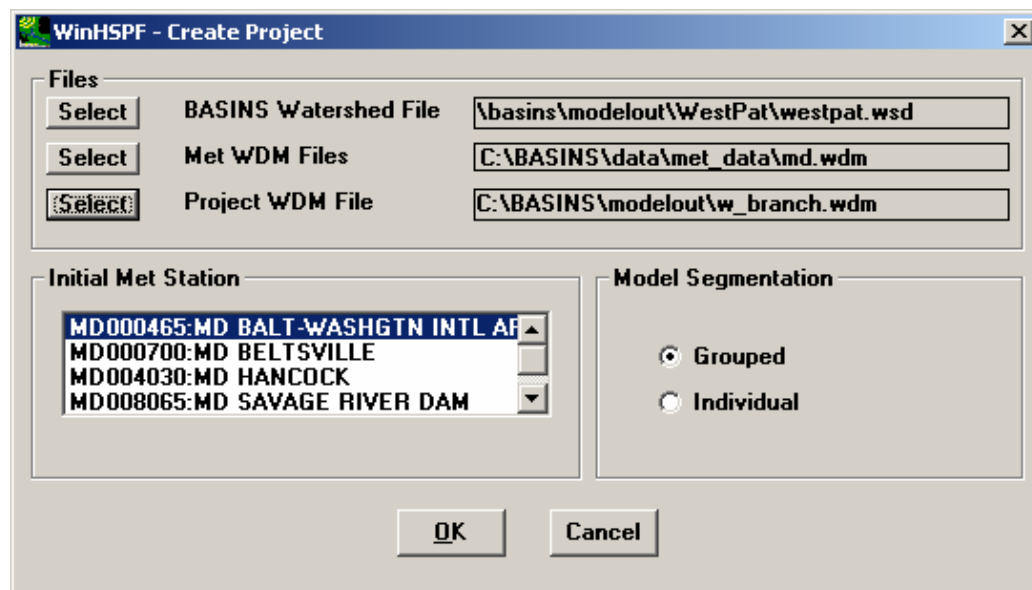
**Note:** Once one or more met WDM files are specified, the “Initial Met Station” list will be populated with the identifier corresponding to met stations available for use in the new HSPF project. One item from this list must be selected in order to have meteorological data included in the HSPF simulation. (After the new project has been created, other met stations may be added through the WinHSPF interface (Duda et al., 2001).)

15. In the “Initial Met Station” section, highlight “MD000465: BALT-WASHGTN INTL AP.”

**Note:** The “Initial Met Station” section contains a list of the Maryland WDM Weather Data Stations. To determine which weather station to use, simply turn “on” and “activate” the WDM Weather Data Stations layer within BASINS, and then use the “Identify” tool to determine which station is closest (or most representative) of your watershed.

The “MD BELTSVILLE” station is actually closer to our watershed than the Baltimore-Washington Airport station is; however, the Beltsville station that is included in the BASINS database has some precipitation data gaps and therefore won’t be used for this particular computer session.

16. Make sure the button beside “Grouped” is selected in the “Model Segmentation” section.

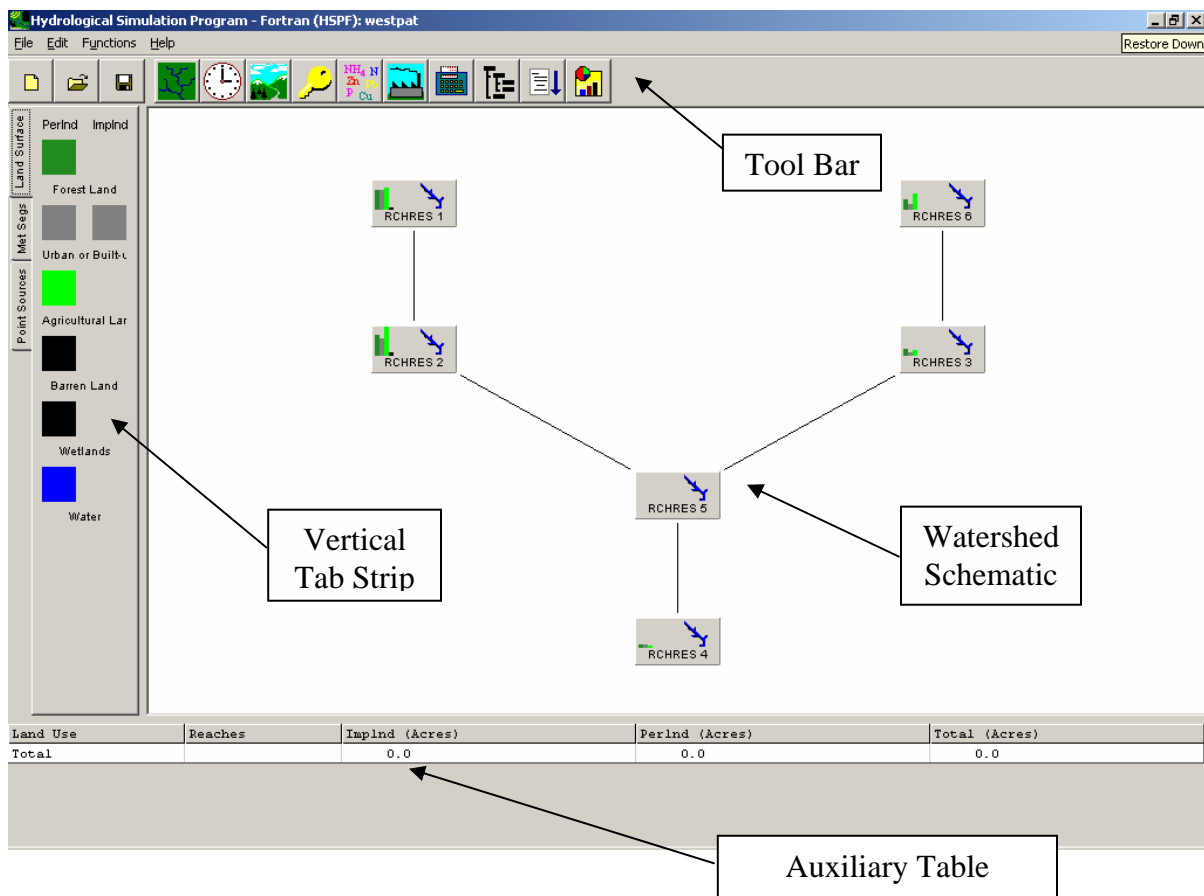


**Note:** In the “Model Segmentation” frame, the “Grouped” option creates a single model segment (one PERLND/IMPLND per land use category) for all collective subbasins, and the “Individual” option creates a single model segment for each subbasin (each subbasin has a group of land use categories).

For example, assume the watershed delineation contains three subbasins, and each subbasin contains a land use called 'Forest'. Using the "Grouped" option will result in one 'Forest' PERLND, (for example, PERLND 101), contributing to each of the three delineated reaches. Selecting the "Individual" option will result in three 'Forest' PERLNDs (for example, PERLND 101, PERLND 201, and PERLND 301) with each of these three PERLNDs contributing to its corresponding delineated reach (Duda et al., 2001).

17. Click OK.

The main window of the HSPF interface (WinHSPF) will open. This window contains several menus, a tool bar, a vertical tab strip, a schematic of the watershed, and an auxiliary table. The vertical tab strip, watershed schematic, and auxiliary table will not be active until a project is active (until a .uci file is opened or a project is created). When WinHSPF is launched from BASINS, the corresponding .uci file will automatically be opened after the appropriate WDM files have been specified. The information in the tab strip, the auxiliary tables, and the watershed schematic are relevant to the open project.













### C. BASINS WinHSPF Tools


*QUESTION ANSWERED:*

4) What tools are available within WinHSPF?





The buttons on the tool bar,          , provide links to the various tools with which the user can interact with the open project. The buttons to the left of the tool bar can be used for creating, opening, and saving projects. The “Run

HSPF” button, , is used to perform the simulation (run HSPF). The other buttons on the tool bar are used to view and modify input values and parameters of the open project and to specify things such as the simulation period and constituents you would like modeled.

The vertical tab strip contains three tabs. The tabs, in conjunction with the figures in the watershed schematic, display information about the project in the auxiliary table. These features can be used to specify whether to display information related to land surfaces, met segments, and point sources. If you highlight items in the tab strip and highlight a figure in the watershed schematic, related data will appear in the auxiliary table. For example, click the “Land Surface” tab and then click the “Forest Land” icon. Click the “RCHRES 2” figure in the watershed schematic. The auxiliary table will show the acres of forested area contributing to Reach 2.

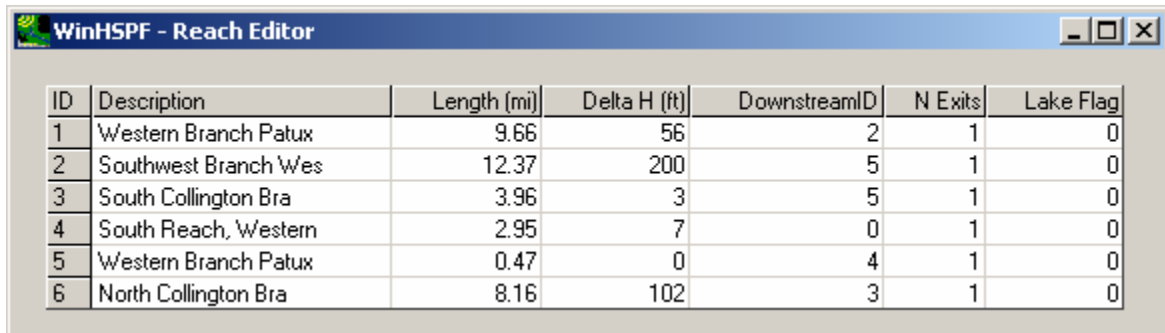
Clicking the “Met Segs” tab indicates which met segments are associated with each reach. The auxiliary table will show information for the selected met segment contributing to the selected reach. Information in the auxiliary table includes the constituent name, WDM data set number, associated multiplication factor for the PERLND/IMPLND segments, associated multiplication factor for the RCHRES segments, and the transformation function used to adjust the time dimension of the data units as necessary (Duda et al., 2001).

The “Point Sources” tab is used to view the point sources that will contribute to the streams in WinHSPF. If you click on a point source, the stream segment to which it will contribute flow and/or constituents will become highlighted in the watershed schematic and data regarding the source will appear in the auxiliary table.

In order to become more familiar with the WinHSPF interface, we will click on each button in tool bar and view the various tools, options, and default input values.

## WinHSPF Reach Editor

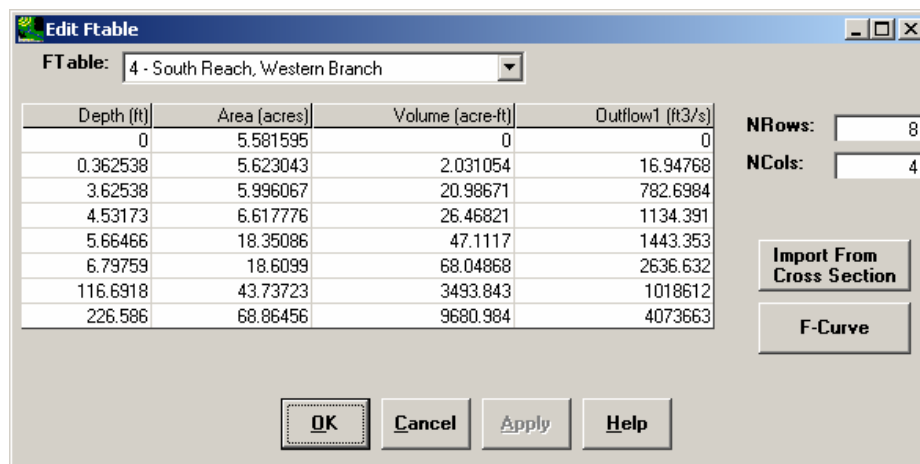
1. Click  (Reach Properties Editor). This editor allows you to modify select properties of each reach. The following screen will appear:



ID	Description	Length (mi)	Delta H (ft)	DownstreamID	N Exits	Lake Flag
1	Western Branch Patux	9.66	56	2	1	0
2	Southwest Branch Wes	12.37	200	5	1	0
3	South Collington Bra	3.96	3	5	1	0
4	South Reach, Western	2.95	7	0	1	0
5	Western Branch Patux	0.47	0	4	1	0
6	North Collington Bra	8.16	102	3	1	0

**Note:** The values in the “Reach Editor” table are automatically calculated when either the manual or automatic delineation tools are used to create the *Streams* layer. If neither of these tools is used, the user must make sure these fields and their corresponding values are included in the attributes of the *Streams* shapefile before launching WinHSPF.

- Click FTABLES. The “Edit Ftable” window will open.



FTable: 4 - South Reach, Western Branch

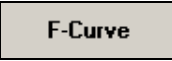
Depth (ft)	Area (acres)	Volume (acre-ft)	Outflow1 (ft <sup>3</sup> /s)
0	5.581595	0	0
0.362538	5.623043	2.031054	16.94768
3.62538	5.996067	20.98671	782.6984
4.53173	6.617776	26.46821	1134.391
5.66466	18.35086	47.1117	1443.353
6.79759	18.6099	68.04868	2636.632
116.6918	43.73723	3493.843	1018612
226.586	68.86456	9680.984	4073663

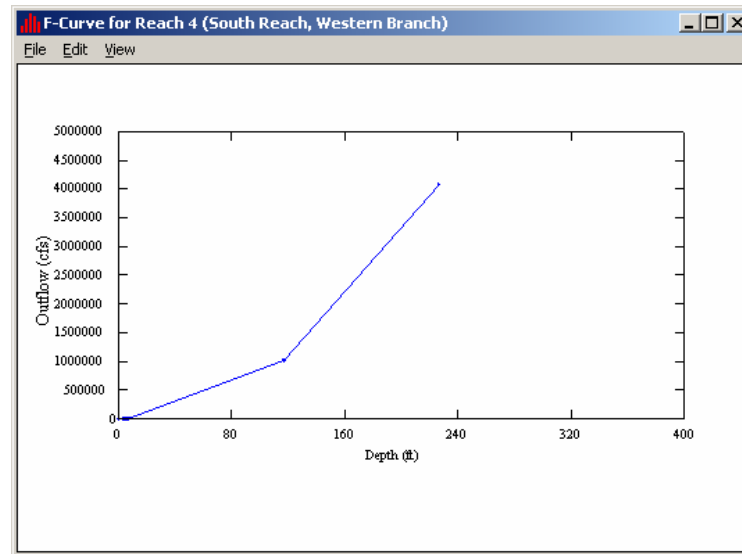
NRows: 8  
NCols: 4

Import From Cross Section  
F-Curve

OK Cancel Apply Help

**Note:** WinHSPF estimates the depth and width of individual reaches using variables such as the area and slope of a subbasin. These estimates are generally inaccurate, but they are better than nothing. It is best to gather data for your watershed’s individual reaches and then use cross-sectional characteristics to populate the F-Tables by either estimating the stage-discharge relationship with an equation such as the Manning’s Equation and then entering them manually or by using the “Import From Cross Section” tool. In this exercise, we will use the default F-tables.

- Click . A graph that shows the relationship between the depth (stage) and reach flow rates will appear.



4. From the **File** menu, select **Close**.

5. Click **Import From Cross Section**.

The dialog box contains a section for 'Cross-Section Files' with 'Open' and 'Save' buttons. Below this is a table titled 'FTable 3' listing various cross-section parameters and their values.

Variable	Description	Value
L	Length (ft)	0.01
Ym	Mean Depth (ft)	0.01
Wm	Mean Width (ft)	0.01
n	Mannings Roughness Coefficient	0.01
S	Longitudinal Slope	0.01
m32	Side Slope of Upper Flood Plain Left	0.01
m22	Side Slope of Lower Flood Plain Left	0.01
w12	Zero Slope Flood Plain Width Left (ft)	0.01
m12	Side Slope of Channel Left	0.01
m11	Side Slope of Channel Right	0.01
w11	Zero Slope Flood Plain Width Right (ft)	0.01
m21	Side Slope Lower Flood Plain Right	0.01
m31	Side Slope Upper Flood Plain Right	0.01
Yc	Channel Depth (ft)	0.01
Yt1	Flood Side Slope Change at Depth (ft)	0.01
Yt2	Maximum Depth (ft)	0.01

At the bottom of the dialog are 'OK', 'Cancel', and 'Help' buttons.


**Note:** Using this editor, you can modify the cross section characteristics or import a table that contains the cross-sectional data (\*.rtf file).

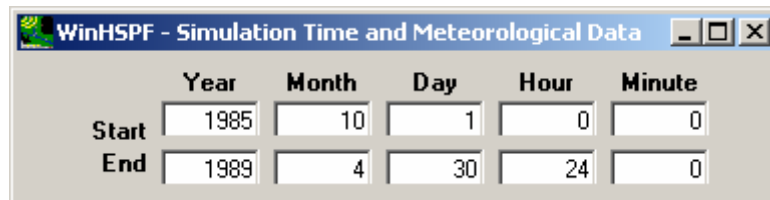
6. Click CANCEL to exit the “Import From Cross-Section” window.
7. Click OK to exit the “Edit FTable” screen.

- Click OK to exit the “Reach Editor” window.

### ***WinHSPF Simulation Time Editor***

A \*.wdm file containing meteorological time series data was specified in the “Create Project” window. In the Simulation Time Editor, you should specify a simulation period (the default time period is for all dates included in your meteorological data file – this time period is usually very long (several decades). The period you specify must be within the time range of data included in your meteorological \*.wdm file.

- Click  (Simulation Time and Meteorological Data).
- Enter the dates shown below for the ‘Start’ and ‘End’ of the simulation.



	Year	Month	Day	Hour	Minute
Start	1985	10	1	0	0
End	1989	4	30	24	0

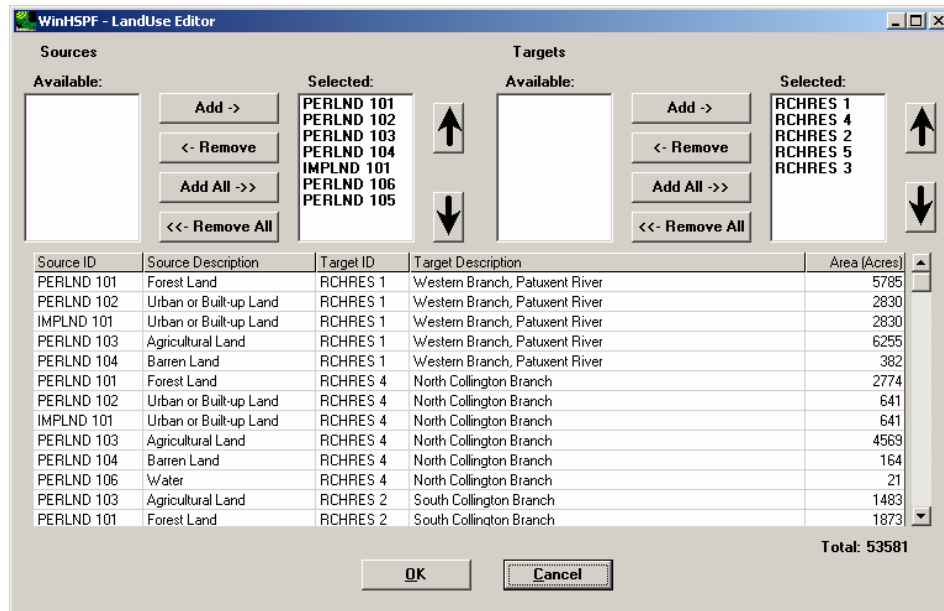
**Note:** The Met Segments section of the Simulation Time and Meteorological Data window contains a grid with a list of the meteorological segments and the operations that they affect. New met segments may be added by clicking ADD button. Click EDIT to modify existing met segments. To edit a met segment, highlight the desired segment in the “Met Seg ID” column then click EDIT. The “Add Met Segment” and “Edit Met Segment” forms are almost identical, except that the Met Segment name in the ‘Add Met Segment’ form may be chosen from a drop-down list. The precipitation data set defines the met segment and therefore, may not be edited in the “Edit Met Segment” form (Duda et al., 2001). We will not add or edit any met segments at this time.

- Click OK.

### ***WinHSPF Landuse Editor***

This tool allows you to edit the area of each land use contributing to each reach and is useful for best management practices and modeling various scenarios. This will be further explained in Exercise 14.

- Click  (Landuse Editor).




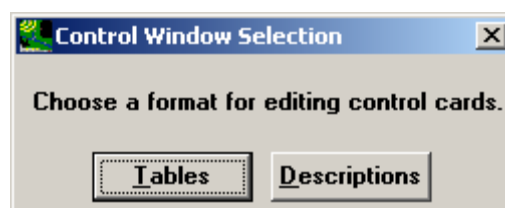
**Note:** The “Sources - Available” box on the left represents the sources and land segments, and the “Targets - Available” box on the right represents the targets and reaches. The table at the bottom of the window lists each “selected” land segment contributing to each “selected” reach. Reaches and land segments are represented in the table if the name of the operation is in the “Selected” box. Sources, land segments, and targets can be moved back and forth between the “Selected” and “Available” boxes. The contents of the grid will automatically be updated.

**Note:** The values in the “Area” column may be edited. The text below the grid will indicate the original total area, the new total area, and the difference between these two. This information allows the user to feel confident that the change was interpreted as desired (Duda et al., 2001).

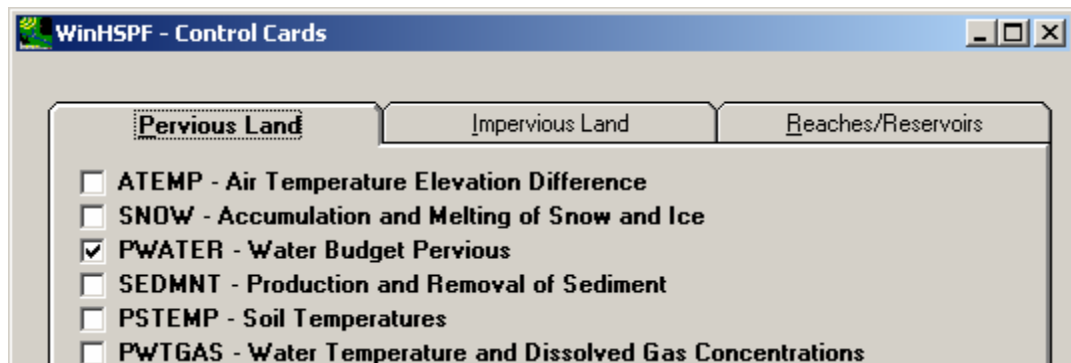
2. Click OK.

### WinHSPF Control Cards Editor

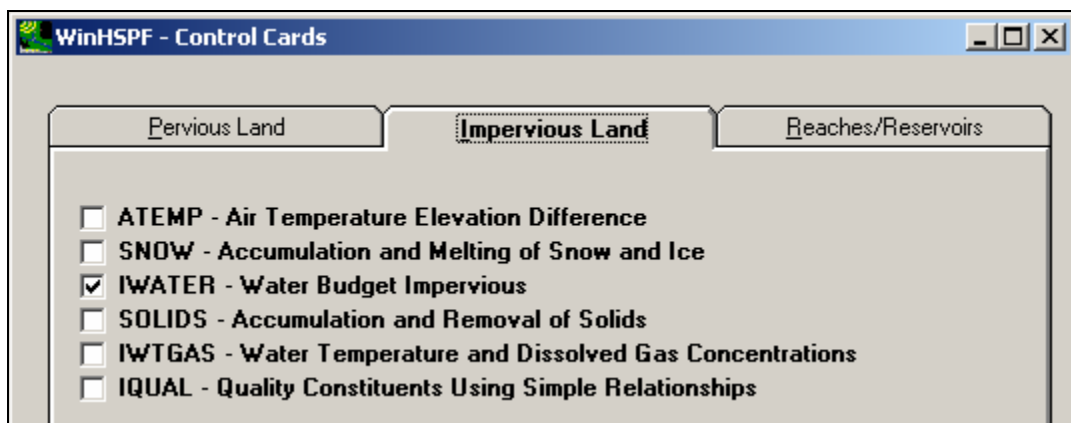
1. Click  (Control Cards).
2. Click DESCRIPTIONS when the following window appears.



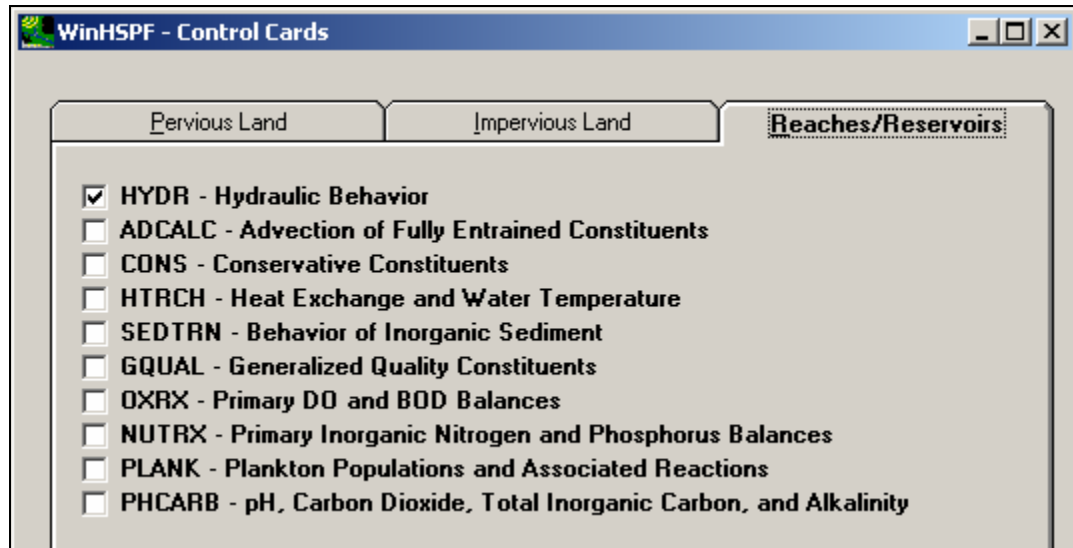
- Note:** This screen allows you to choose which parameters you would like to model in WinHSPF. Each tab (PERLND, IMPLND, and RCHRES) contains a list of corresponding modules and parameters. You can specify which parameters and modules you want to model by checking the boxes beside the parameters. For example, if you wanted to model the accumulation and melting of snow and ice, you would check the box beside “Snow.” When you turn “on” (check the box beside) a module through this editor, other sections might turn on automatically because there are rules in the model’s coding that require certain modules as prerequisites to others. These operating rules are based on the guidelines in *BASINS Technical Note 3: the NPSM/HSPF Simulation Module Matrix*. Following these operating rules, all required and recommended prerequisite sections are turned on when appropriate (Duda et al., 2001).
3. The “Pervious Land” tab will be displayed. Make sure that “PWATER” (water budget pervious) is selected.



4. Click the “Impervious Land” tab.



5. Make sure that “IWATER” (water budget impervious) is selected.
6. Click the “Reaches/Reservoirs” tab.




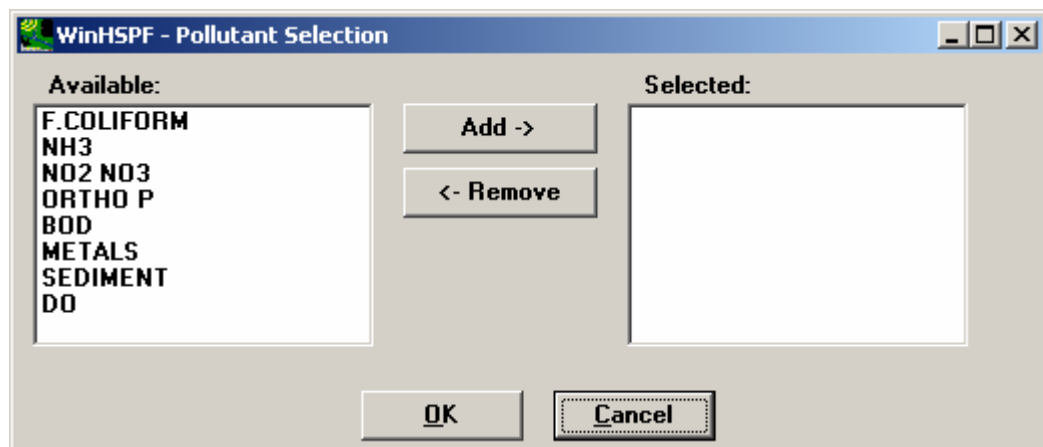
7. Note that “HYDR” (hydraulic behavior) is selected, and select “ADCALC.”

**Note:** Notice that PWATER is selected in the Pervious Land tab, IWATER is selected in the Impervious Land tab, and HYDR is selected in the Reaches/Reservoir tab. These three selections indicate that we will be modeling overland flow in pervious and impervious land segments and it will then be routed into the stream reach.

8. Click OK.

### Pollutant Selection


1. Click  (Pollutant Selection). In this window, we can specify which water quality constituents we would like to model using HSPF. We will not change anything in this screen since we are not modeling water quality in this session.

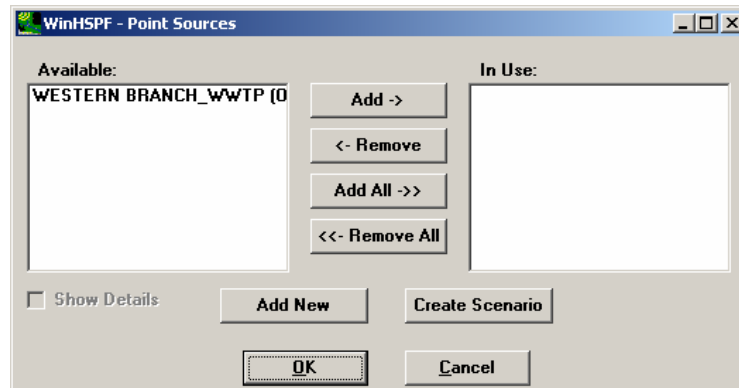


2. Click OK.

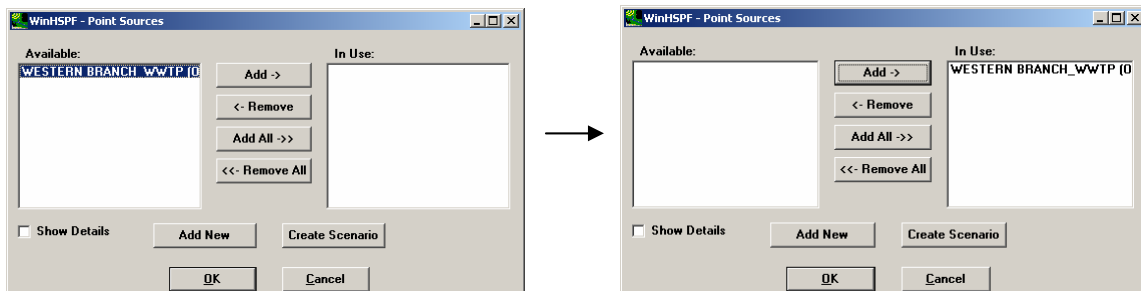
### Point Source Editor

This tool is used to view and manage point sources in the simulation. We will add the PCS point source that contributes to Reach 4.

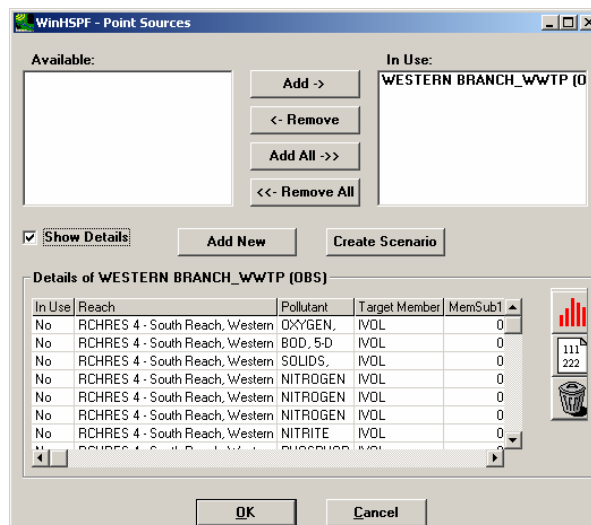
1. Click  (Point Sources). The “Point Sources” window will open.



2. In the “Available” box, select “WESTERN BRANCH WWTP (OBS)” and click “Add->.” The record will now be in the “In Use” box.

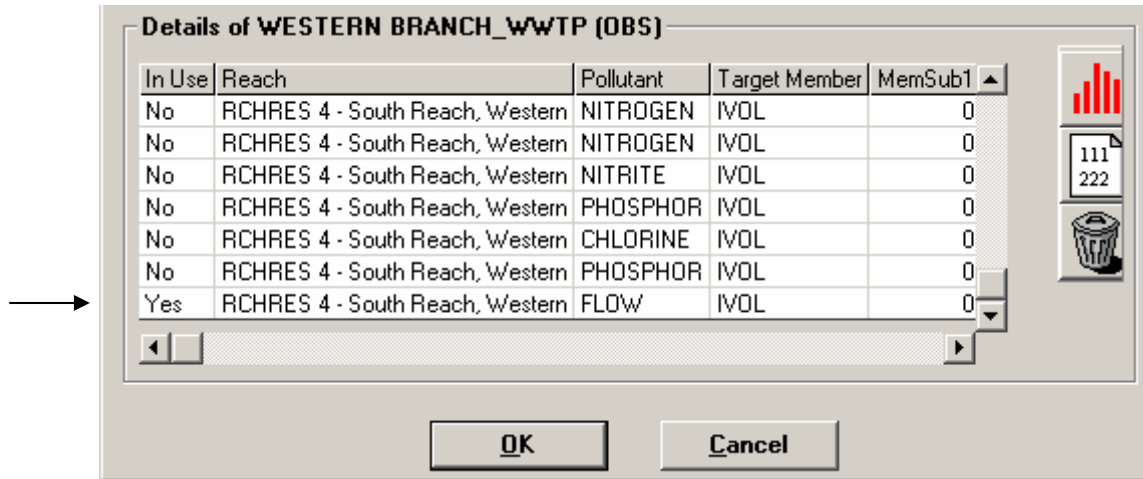


3. Click in the box next to “Show Details.” The Point Sources editor will expand.



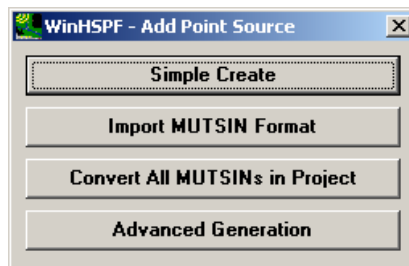


4. The list under “Details of WESTERN BRANCH\_WWTP(OBS)” shows the pollutants included in the PCS file. The model will not simulate a constituent unless its corresponding “In Use” box is set to “Yes.” Scroll down until you see “FLOW” in the “Pollutant” column. Double click the corresponding “In Use” box and select “Yes.”



5. Click OK.

**Note:** If you want to use point source data other than that provided by BASINS, click ADD NEW. The following window will appear.



The following gives explanations for the buttons on the “Add Point Source” window.

*Simple Create:* This tool allows you to add point source data by specifying a scenario name, a reach to which to add the point source, the pollutant(s), a facility name, and a daily load.

*Import MUTSIN Format:* This allows you to select a Mutsin file that contains the point source information. The user must specify a scenario name, a reach to which to add the point source, the pollutant(s), and a facility name.

*Convert All MUTSINs in Project:* This tool scans the HSPF input sequence (the UCI file) and converts any found Mutsin files to WDM data sets in the project WDM file.


*Advanced Operation:* This tool allows you to modify the time series in your point source data file.

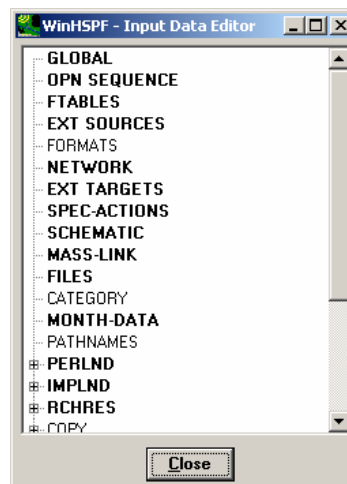
**Note:** You might want to edit values in one of the point source data sets. You can do so by highlighting a field in the row associated with the desired data set and

then clicking the “List/Edit Selected Point Sources” button, . Also, you can view the results graphically by clicking the “Graph Selected Point



Sources” button, . You can delete a point source by clicking the “Delete Selected Point Sources” button, .

### *WinHSPF Data Editor*

1. Click  (Data Editor). The following window will appear.



**Note:** The “Input Data Editor” window contains a tree diagram of all the possible blocks and tables of HSPF. The tree diagram is navigated by clicking on an item name. **Double clicking** on an item name reveals the sub-items within that item if any exist.

2. Click  next to “PERLND.”
3. Click  next to “PWATER.”
4. **Double click** “PWAT-PARM2.”

Edit PERLND:PWAT-PARM2

☒ Show Description

OpNum	Description	FOREST	LZSN	INFILT	LSUR	SLSUR	KVARY	AGWRC
101	Urban or Built-up Land	0	6	0.16	350	0.013207	0	0.98
102	Agricultural Land	0	6	0.16	350	0.013207	0	0.98
103	Forest Land	1	6.5	0.16	350	0.013207	0	0.98
104	Barren Land	0	6	0.16	350	0.013207	0	0.98
105	Wetlands/Water	1	6.5	0.16	350	0.013647	0	0.98

Table: PWAT-PARM2, Second group of PWATER Parameters.  
 Parameter: AGWRC is the basic groundwater recession rate if KVARY is zero and there is no inflow to groundwater (rate of flow today/rate yesterday).

\*\*\* < PLS>    FOREST       LZSN       INFILT       LSUR       SLSUR       KVARY       AGWRC  
 \*\*\* x   - x            (in)       (in/hr)       (ft)                (1/in)       (1/day)

OK Cancel Apply Help

**Note:** This editor is used to modify site-specific and calibration parameters varied in model calibration. Notice that as you click in each column corresponding with a parameter, the definition of that parameter is displayed in the text box below the table.

- Click **HELP**.

HSPF Manual

Hide Back Print Options

Contents Index Search

PWATER input

- PWAT-PARM
- PWAT-PARM
- PWAT-PARM
- PWAT-PARM
- PWAT-PARM
- PWAT-PARM
- PWAT-PARM
- PWAT-PARM
- MON-INTERC
- MON-UZSN
- MON-MANNIN
- MON-INTERF
- MON-IRC
- MON-LZETP
- PWAT-STATI
- IRRIG-PARM
- IRRIG-PARM
- CROP-DATE
- CROP-STAGE
- CROP-SEAS
- SOIL-DATA
- SOIL-DATA2
- SOIL-DATA3

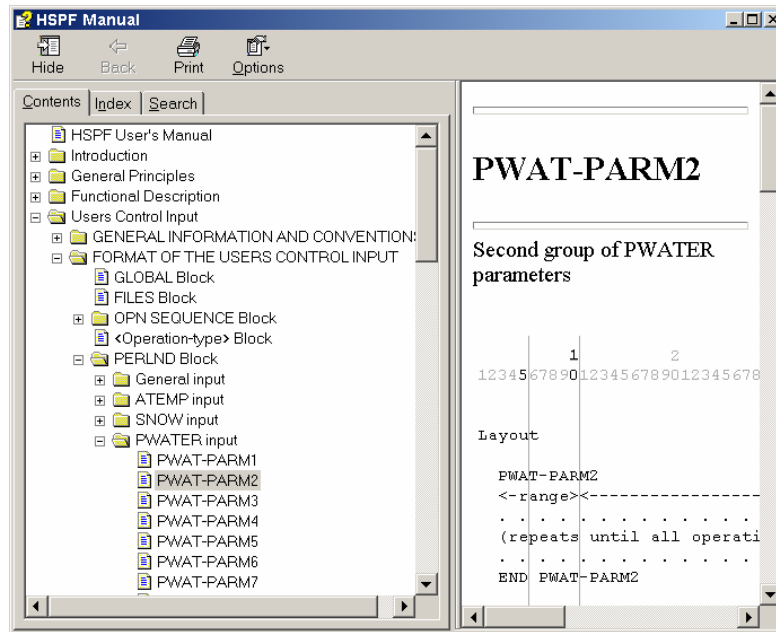
## PWAT-PARM2

### Second group of PWATER parameters

1					2					3					4				
1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890					
Layout																			
PWAT-PARM2																			
<-range><-----pwatpar																			
.....																			
(repeats until all operations of this type																			
.....																			
END PWAT-PARM2																			
Example																			

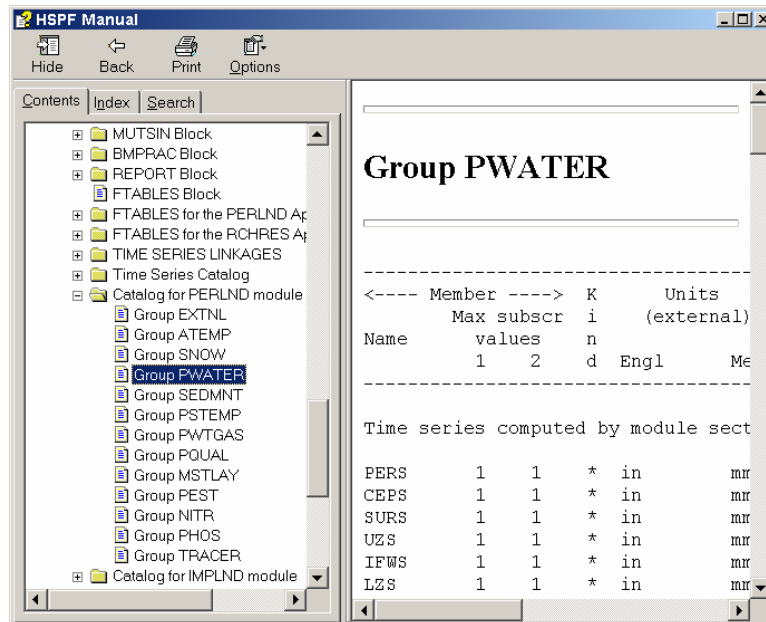
**Note:** Notice that the HSPF Manual opens to the page corresponding to the table PWAT-PARM2. This page of the manual provides additional details about the individual parameters in the PWAT-PARM2 table, including definitions and limits.

- Scroll up to the top of the “Contents” at the left side of the manual.



**Note:** The tree diagram in the “Contents” shows the general format of the HSPF manual. The “Functional Description” section contains detailed explanations of the algorithms and scientific basis of HSPF, while the “Users Control Input” section contains operational details.

7. Scroll down in the “Contents.” Within the “User Control Input” section, open the ‘Catalog for PERLND module’ folder, and click on the ‘Group PWATER’ page.




**Note:** This page is known as the “Time Series Catalog” for PWATER. The “Time Series Catalog” lists all time series computed by each HSPF module, including the units of the output. It also lists the possible input time series for each

module. The list of input time series indicates either prerequisite sections that must be turned on or time series that must be supplied externally (such as through the input WDM file) for a particular module section.

8. Browse the HSPF Manual as much as you wish. When you are finished, close the manual by clicking the 'X' in the top right corner.
9. Toggle to WinHSPF. Click OK or CANCEL in the "Edit PERLND:PWAT-PARM2" window.
10. Click **CLOSE**.

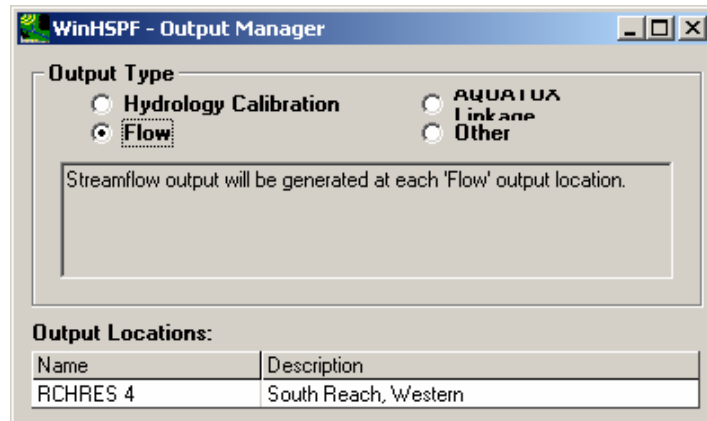
### *WinHSPF Output Manager*

1. Click  (Output Manager). The Output Manager allows you to select what will be written as the output time series. For each output type, you can choose one or more locations for which output will be generated.

**Note:** The first output type is Hydrology Calibration. This button will already be selected when entering the Output Manager. With this button selected, the list below the radio buttons displays the HSPF calibration locations within the current HSPF project. Under the buttons is a box explaining which output time series will be generated during the HSPF model run.

You can add calibration locations to this list by clicking ADD. A window will appear that lists all available calibration locations (i.e., the reaches of the watershed) along with two text fields. The user must choose one of the calibration locations, and then enter an eight-character identifier for that location. This identifier is used as the location ID attribute in the WDM time-series data sets that will be created. The user may also enter a "base data set" number for the data sets to be created. The new data sets will be numbered consecutively as the available data sets following the base number. In this section, we are modeling only flow, so we will only specify output locations for "Flow."

2. Click the radio button beside "Flow."

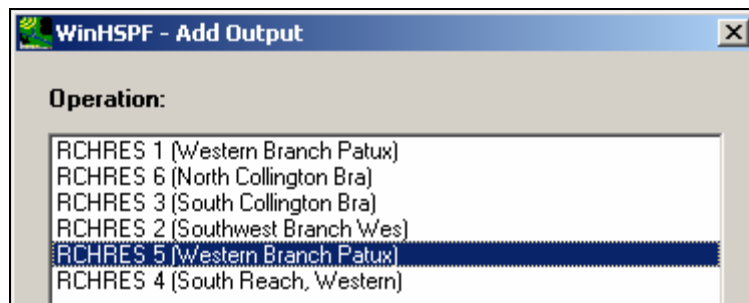


**Note:** Notice that “RCHRES 4” is the only stream listed in the output locations. We would like to look at the flow in “RCHRES 5” because there is a flow gage located on Reach 5.

3. Highlight “South Reach, Western” in the “Description” text box and click REMOVE.
4. Click YES when the following message appears.



5. Click ADD.
6. Highlight “RCHRES 5 (Western Branch Patux).”




7. Click OK. Notice that “RCHRES 5” is now listed as an output location. This means that we will be able to view the simulated flow data for Reach 5.
8. Click CLOSE to exit the “Output Manager.”

## D. Running WinHSPF

In this section, we will run our WinHSPF model for the West Branch of the Patuxent River.

*QUESTION ANSWERED:*

4) *How do I run my WinHSPF model?*

1. Click  (Run).
2. If the following message appears and asks if you want to save changes that have been made, click **SAVE/RUN**. Your *.uci* file will be saved and the model simulation run will begin.



**Note:** A screen will appear, indicating that the input file is being processed and run. When this disappears, the model run is complete.

**Note:** We would like to compare the simulated flow for Reach 5 (recall that we specified RCHRES 5 as a “Flow” output location) with observed flow in Reach 5. Before we do this, we need to add the gage data to the current project.

3. Close the WinHSPF window.

**IMPORTANT NOTE:** IF YOU DO NOT CLOSE THE WINHSPF WINDOW, YOU WILL NOT BE ABLE TO COMPLETE SECTION E.

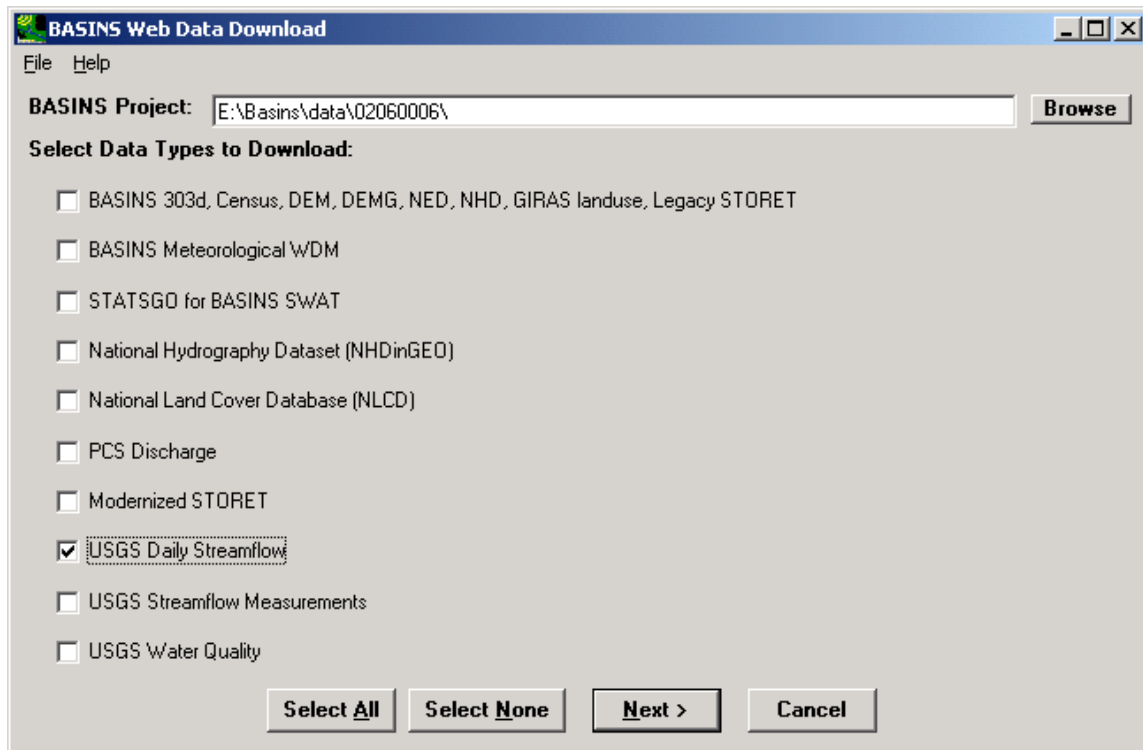
## E. Adding Gage Data To GenScn

*QUESTION ANSWERED:*

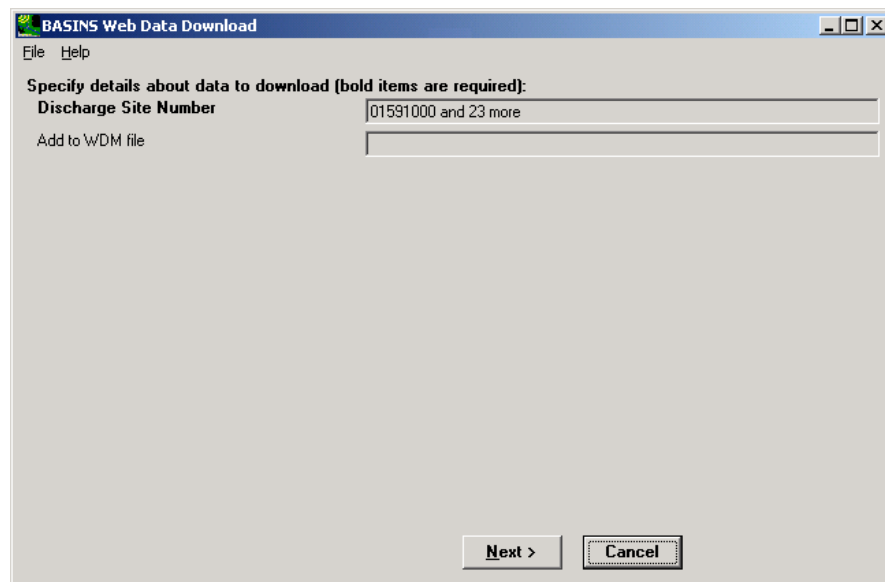
5) *How do I add gage data to my WinHSPF project file?*

In this section, we will download the USGS gage data for Reach 5 and add it to the *w\_branch* project *\*.wdm* file using WDMUtil. If you are unable to connect to the internet, please skip to step 12.

1. Within BASINS, from the **File** menu, select **Download Data**.
2. Check the box next to “USGS Daily Streamflow.” Make sure no other data sets are selected. Your window should look like the following.



3. Click NEXT. The following window will appear.



4. Click in the “Discharge Site Number” box to specify which site to download. A list of discharge sites should appear.
5. Scroll down until you find discharge site “01594526” with a station name of “Western Branch at Upper Marlboro, MD” and click on that row.



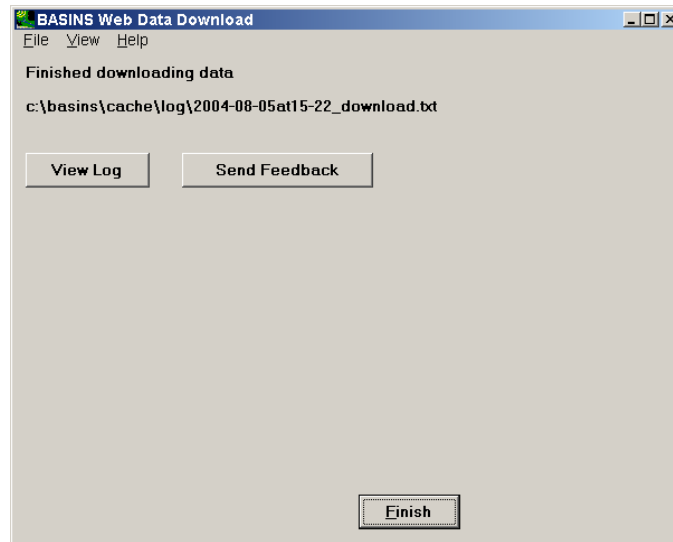
discharge_sit	station_nm	state_	county_	huc_cc	dec_lat_	dec_long	coord_acy	dec_lat_long_dat	al
01594670	HUNTING CREEK NEAR HUNTINGTOWN, MD	24	009	020600	38.58400	-76.60523	S	NAD83	10
01594400	DORSEY RUN NEAR JESSUP, MD	24	003	020600	39.12094	-76.78302	S	NAD83	12
01591500	CATTAIL CREEK AT ROXBURY MILLS, MD	24	027	020600	39.25482	-77.04497	S	NAD83	37
01594710	KILLPECK CREEK AT HUNTERSVILLE, MD	24	037	020600	38.47706	-76.73524	S	NAD83	50
01594800	ST LEONARD CREEK NEAR ST LEONARD, MD	24	009	020600	38.44891	-76.49513	S	NAD83	5
01594526	WESTERN BRANCH AT UPPER MARLBORO, MD	24	033	020600	38.81422	-76.74872	S	NAD83	30
01594600	COCKTOWN CREEK NEAR HUNTINGTOWN, MD	24	009	020600	38.64095	-76.63496	S	NAD83	40

6. Click USE SELECTED.

**Note:** In order to compare the simulated data to the observed data, we need to add the gage data to a WDM file. Therefore, in the following steps we will add the gage data to the project WDM file (*w\_branch*) that we specified in the “WinHSPF Create Project” window.

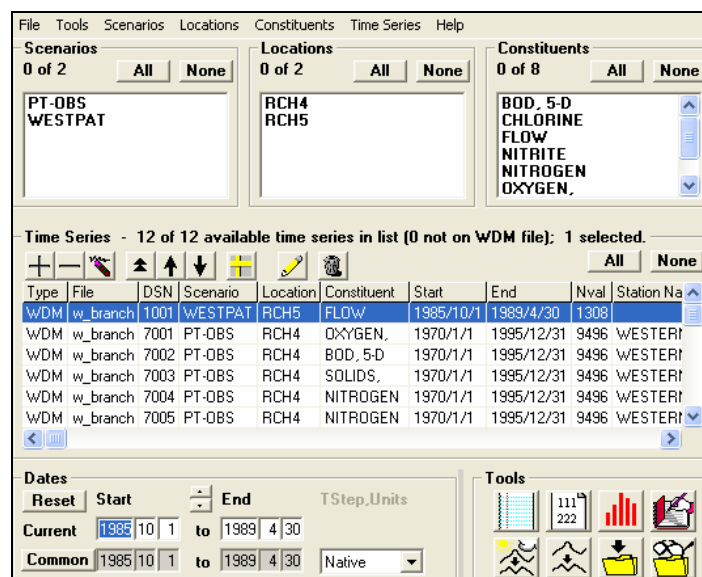
7. Click in the box next to “Add to WDM File.”
8. Navigate to *c:\basins\modelout\* and click on “*w\_branch.wdm*.”
9. Click OPEN. Your window should look like the following:

10. Click NEXT.
11. Click FINISH when the following window appears. The gage data you specified has now been added to your *w\_branch.wdm* file.



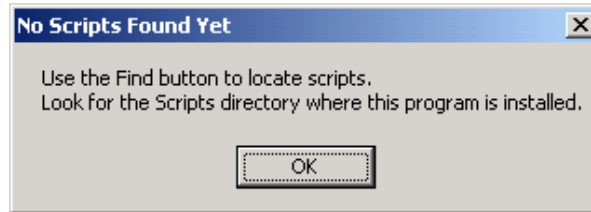
If for some reason you were unable to download the data through the **Data Download** tool, the downloaded data has been provided for you. You may import this data into the *w\_branch.wdm* file using WDMUtil by going through the following steps. **If you were able to import the data successfully, you may skip steps 12 through 29 below and continue with step 30.**

12. From the **Start** menu under **Programs**, select **BASINS** and then **WDMUtil**.
13. From the **File** menu, select **Open**.
14. Navigate to *c:\basins\modelout\* and click on “*w\_branch.wdm*.”
15. Click OPEN. The WDM will open:

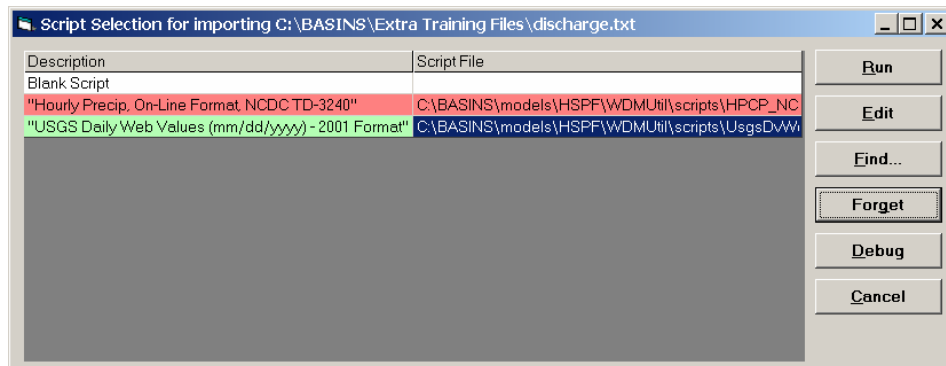


16. From the **File** menu, select **Import**.

17. Navigate to *c:\basins\extra training files\* and select “*discharge.txt*.” Click OPEN.
18. Click OK if the following message box appears.

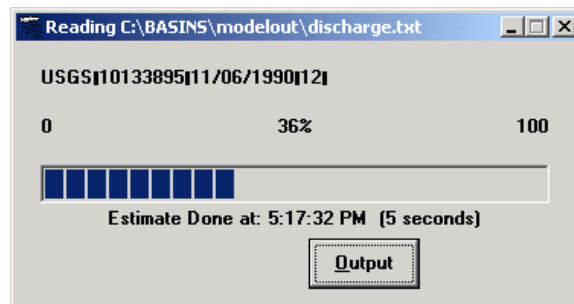


19. The “Script Selection” window will open. Click FIND.
20. Navigate to *c:\basins\models\hspf\WDMUtil\scripts* and select “*UsgsDvWeb\_MDY\_2001.WS*.”
21. Click OPEN.
22. Under script file, select the record that corresponds to “USGS Daily Web Values (mm/dd/yyyy) – 2001 Format.” It will turn blue.



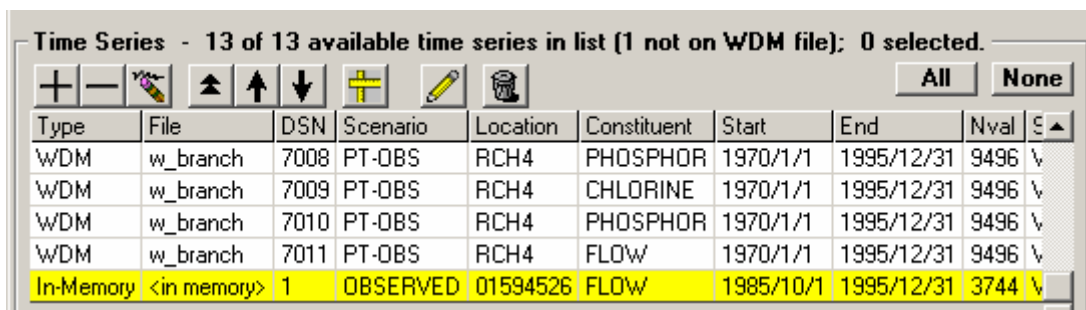
23. Click RUN.

The “Script Selection” window will disappear and a run window will open, indicating that the discharge data is being processed and added to the current WDM project.



When this process is complete, a new record will appear in the time series list. It will be highlighted (yellow).

Time Series - 13 of 13 available time series in list (1 not on WDM file); 0 selected.



Type	File	DSN	Scenario	Location	Constituent	Start	End	Nval	Σ
WDM	w_branch	7008	PT-OBS	RCH4	PHOSPHOR	1970/1/1	1995/12/31	9496	✓
WDM	w_branch	7009	PT-OBS	RCH4	CHLORINE	1970/1/1	1995/12/31	9496	✓
WDM	w_branch	7010	PT-OBS	RCH4	PHOSPHOR	1970/1/1	1995/12/31	9496	✓
WDM	w_branch	7011	PT-OBS	RCH4	FLOW	1970/1/1	1995/12/31	9496	✓
In-Memory	<in memory>	1	OBSERVED	01594526	FLOW	1985/10/1	1995/12/31	3744	✓

24. Click the new time series. It will turn blue when selected.


25. In the “Tools” frame, click  (Write Time Series to WDM).

26. In the Output DSN text box, type “27.” Leave other fields as they are.

Specify Output Data-set Number(s):  
Select/Enter Scenario, Location, Constituent as needed;  
Click Write button to store data on WDM file.

☒ Use common period for all data sets, as defined on main form  
☐ Use full period for each data set

DSN/ID	Output DSN	Scenario	Location	Constituent
1		OBSERVED	01594526	FLOW

27. Click WRITE.
28. Click OK when you are told that the file was stored successfully.
29. From the **File** menu, select **Exit**.
30. From the **Start** menu under **Programs**, select **BASINS** and then **WinHSPF**.
31. From the **File** menu, select **Open** or click the “Open Existing Project” button, .
32. Navigate to *c:\basins\modelout\Westpat\* and select “westpat.uci.” Click OPEN.

## F. Viewing WinHSPF Output in GenScn

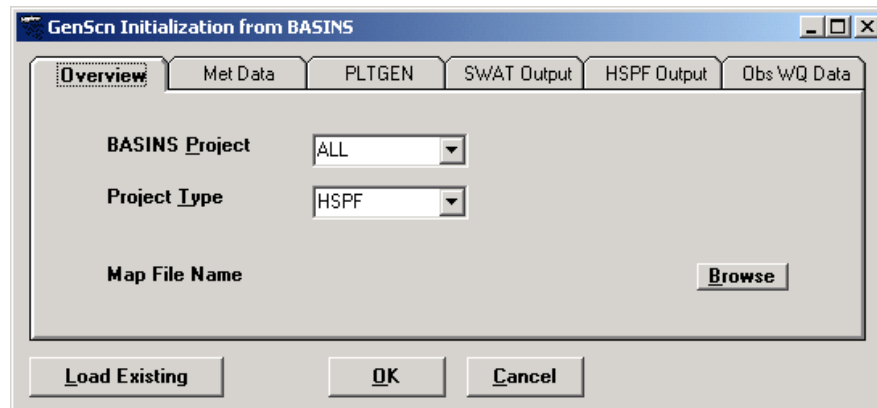
*QUESTION ANSWERED:*

6) How do I use GenScn to view my simulation results?

In this section, we will view the results of the simulation using the GenScn postprocessor. GenScn was designed to manage large volumes of input, output, and calibration data used by and produced in models such as SWAT and WinHSPF. This software has many different applications; however, in this exercise we will focus on those necessary for viewing the output from WinHSPF and corresponding gage data.

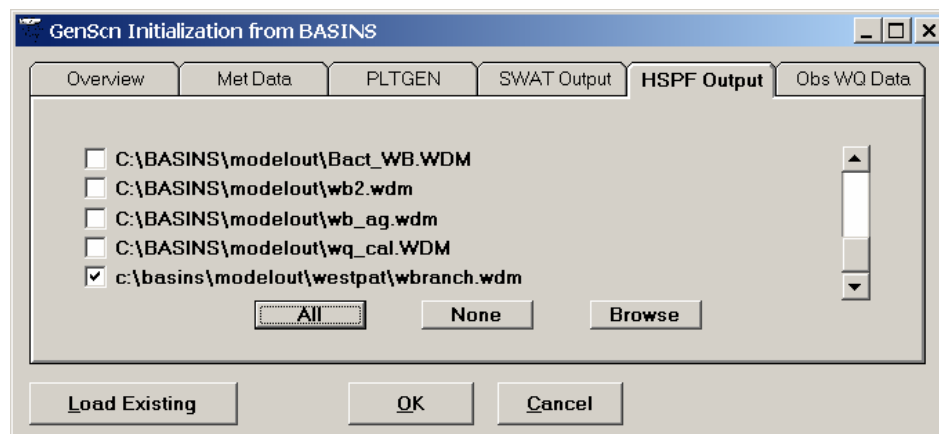
1. Click  (View Output button).

**Note:** The “GenScn Initialization from BASINS” window will appear. This form has six different tabs because many types of files can be viewed within the GenScn interface.



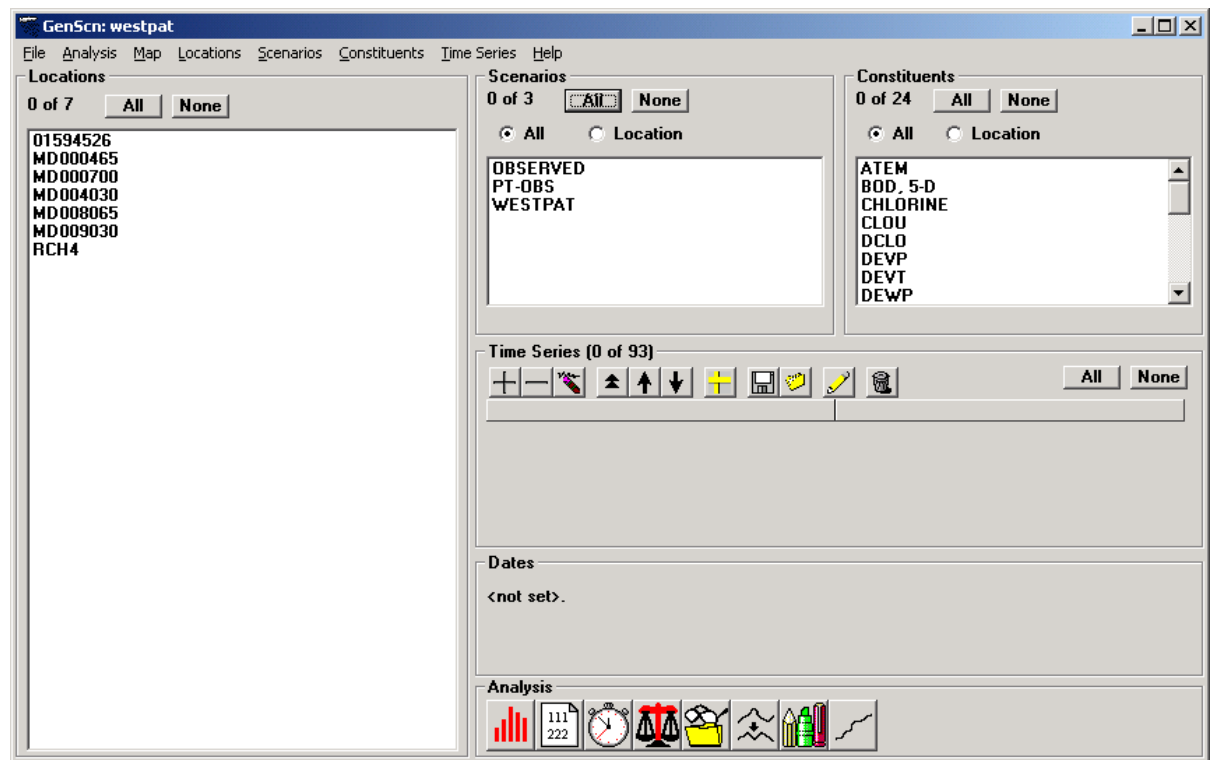
**Note:** The “Overview” tab shows information about the GenScn project that is being created from the BASINS interface.

2. Click the “HSPF Output” tab and check the box beside *c:\BASINS\modelout\w\_branch*. Click OK.



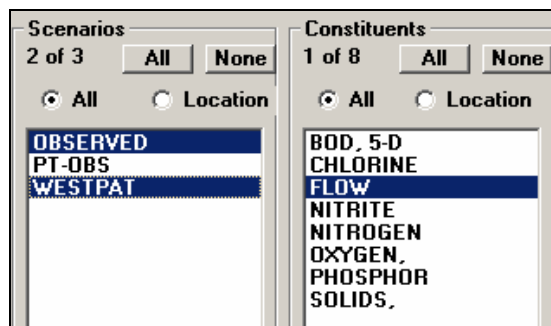
The GenScn window will open. This window has six sections: Locations, Scenarios,


## Constituents, Time Series, Dates, and Analysis.



Recall that we want to look at a graph that includes both the model's flow simulation values and observed flow values for the same reach in order to ascertain how well we are simulating flow in Reach 5. We will graph both times series in the following steps.

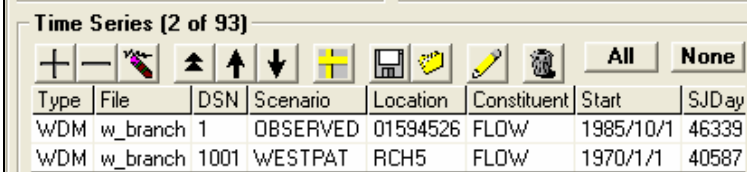
3. In the "Scenarios" frame, select "WESTPAT" and "OBSERVED."
4. In the "Constituents" frame, select "FLOW."




5. In the "Time Series" frame, click  (Add to Time Series List). All time series for flow within the "WESTPAT" and "OBSERVED" scenarios will be listed in the Time Series section.

6. In the “Time Series” frame, highlight the record that corresponds to the scenario name “WESTPAT.” Press SHIFT and highlight the “OBSERVED” time series.

Time Series [2 of 93]



Type	File	DSN	Scenario	Location	Constituent	Start	SJDay
WDM	w_branch	1	OBSERVED	01594526	FLOW	1985/10/1	46339
WDM	w_branch	1001	WESTPAT	RCH5	FLOW	1970/1/1	40587

7. In the “Analysis” frame, click  (Generate Graphs).
8. Make sure the “Standard” box is checked.

Graph

Select desired plots, then press Generate.

☐ Multiple WQ Plots

☒ Standard

☐ Residual (TS2 - TS1 vs. time)

☐ Cumulative Difference vs. time

☐ Bar Chart

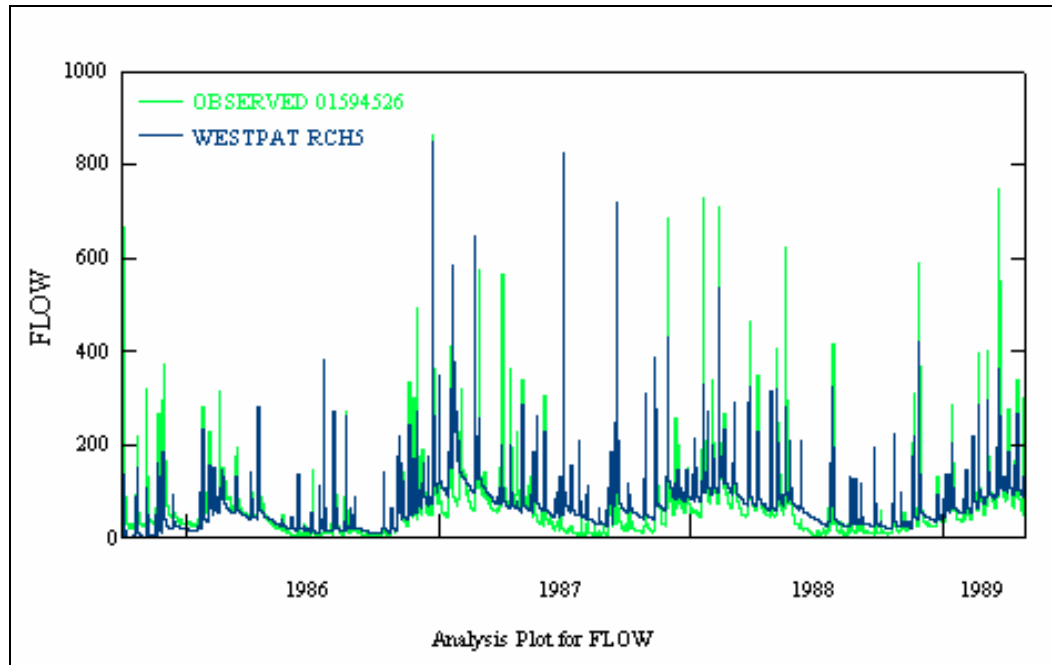
☐ Flow/Duration

☐ Difference (TS2 - TS1) vs. TS1

☐ Scatter (TS2 vs. TS1) ☐ 45 deg/regress lines

☒ Current Period ☐ Storms

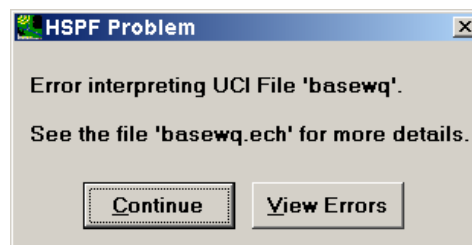
9. Click GENERATE. The plot window will open:



10. After looking at the graph, exit the Genscn Standard Plot. Click CANCEL to exit the "Graph" window.
11. Zoom in on smaller time periods by changing the start and end dates in the "Dates" frame. Repeat steps (6-9) to look at the graph of this time period.
12. Exit any open graphs and close GenSCN. You don't need to save any changes.

### Special Note about HSPF Errors in WinHSPF:

From time to time changes made to the User Control Input (\*.uci) file through WinHSPF (or a text editor) are interpreted as errors the next time the \*.uci is opened in WinHSPF. This condition is indicated by a message similar to the following when opening a \*.uci file in WinHSPF:



Details of the error encountered are written to the file known as the "Echo" file which is named with the same base name as the \*.uci file only with a \*.ech extension (e.g., *w\_branch.uci* will have an "Echo" file named *w\_branch.ech*). Clicking the "View Errors" button will provide a view of the \*.ech file at the point of the first error. HSPF error



messages are enclosed in asterisks as shown below:

```

WinHSPF Error View of basewq.ech

COPY      100
END INGRP
END OPN SEQUENCE

*****
*
* ERROR/WARNING ID: 211 23
*
* The text ' PER&ND 191' is not recognized as a heading,
* operation-type or delimiter, and thus will be ignored. Is it
* misspelled or placed in the wrong fields?
*
*****

FINISHED PROCESSING OPN SEQUENCE BLOCK
=====

PROCESSING FTABLES BLOCK

USERS CONTROL INPUT IS:
  FTABLES

  FTABLE 680

<< < 4 > >> of 100  FORTRAN Outp  Close  Print  Find
  
```

The \*.ech file may also be opened in any text editor (such as Wordpad). Searching for the text 'Error' will take the focus to the portion of the file containing error messages. Once the error messages are found, the cause of the problems can be identified and resolved.

It should also be noted that it is also possible to encounter errors by HSPF during simulation that prevent the model from running to completion. If the simulation does not appear to run to completion (for example the output time series plots do not contain data for the full span of the run), the \*.ech file should be examined for error messages. The \*.ech files are generally located in the same directory as your \*.uci file.

### Reference:

Duda, P., J. Kittle, Jr., M. Gray, P. Hummel, R. Dusenbury. *WinHSPF Version 2.0: An Interactive Windows Interface to HSPF (WinHSPF), User's Manual*. Washington, D.C.: U.S. Environmental Protection Agency, 2001.